Tridimensional model coupling using Schwarz methodology - Application to a water intake of a hydroelectric plant

Mehdi Pierre Daou, Olivier Bertrand, Eric Blayo, Antoine Rousseau

TRIDIMENSIONAL MODEL COUPLING USING SCHWARZ METHODOLOGY - APPLICATION TO A WATER INTAKE OF A HYDROELECTRIC PLANT

Mehdi Pierre DAOU[1]
Mehdi-pierre.daou@arteliagroup.com

Olivier BERTRAND [1]
Olivier.BERTRAND@arteliagroup.com

Eric BLAYO[2]
eric.blayo@univ-grenoble-alpes.fr

Antoine ROUSSEAU[3]
antoine.rousseau@inria.fr

KEY WORDS

Numerical modeling, coupling method, water intake**,** Interfoam, Telemac-3D

ABSTRACT

Progress has been performed for decades, in terms of physical knowledge, numerical techniques and computer power, that allows to address more and more complex simulations. For many applications, engineers have to implement complex "modeling systems", coupling several models and softwares, representing various parts of the physical system. We address in this article model coupling techniques using Schwarz's methodology, which is based on domain decomposition methods. The basic principle is to reduce the resolution of a complex problem into several simpler sub-problems, thanks to an iterative algorithm. These methods are particularly well suited for industrial codes, since they are very few intrusive. We firstly discuss a coupling test case between two monophasic/diphasic tridimensional models, using Telemac-Mascaret and OpenFOAM software systems. Secondly, we analyze such a coupling for an operational engineering study of a water intake for a hydroelectric plant and a short comparison with physical model results.

1. INTRODUCTION

Fluid dynamics studies, either for industrial or geophysical flows, become more and more complex, multidisciplinary and require numerical modeling tools which must be efficient and interoperable. For numerous applications (impact studies or flood prevention for instance), it may be necessary to

[1]ARTELIA Eau & Environnement, 6 rue de Lorraine, 38130 Echirolles, France

[2] Université Grenoble Alpes

[3] Inria & IMAG UMR 5149, Team LEMON

design a modeling system that couples processes representing different parts of the physical problem. These models may differ in several ways, related either to the physics and/or to the numerics.
For instance, it is rather frequent that the heterogeneity of the physics phenomenae along a flow is such that multi-model approach is interesting to solve the different physical scales. In this context, the present work addresses the question of coupling models techniques using Schwarz's methodology, which is a domain decomposition method, between a monophasic model (free surface flow) and a diphasic model (solving simultaneously air and water).
We firstly discuss the coupling problem between both models, using Telemac-Mascaret and OpenFOAM software systems.We will describe the governing equations of these models, then we will see how to use Schwarz algorithm and we will analyze a simple test-case.
This coupling enables to solve complex flows, that the Telemac-3D solver alone cannot address (breaking waves, water blade, closed-conduit flow, etc.), by locally using InterFOAM where necessary (InterFOAM is very expensive in terms of computations). We will end up with an analyze of such a coupling for an operational engineering study of a water intake for a hydroelectric plant and a short comparison with physical model results.

2. METHODOLOGY OF COUPLING (SCHWARZ METHOD)

We will deal all along this paper with the coupling of a 3-D Navier-Stokes (NS3D) model dedicated to free surface flow (monophasic flow) and a diphasic 3-D Navier-Stokes model. Such interaction occurs for instance in river modeling with engineering structures (like weir, hydroelectric plant…). The coupling interface must be located in an area where the flow is only free surface (river or open-channel).
Firstly, we will describe the governing equations in each model and the coupling interface conditions. Secondly, we will study a test-case (a U-shaped channel) to analyze how to use the coupling algorithm, and discuss numerical results.

2.1 Governing equations

2.1.1 3-D Navier-Stokes model

In the Telemac-Mascaret software, we use the 3-D solver allowing to resolve NS3D model for free surface flow (Hervouet 2007).
The flows are considered as incompressible with a Newtonian fluid (water). These hypotheses will be also valid for diphasic model.
The NS3D equations system is so :

$$\begin{cases} \nabla \cdot \boldsymbol{U} = 0 \\ \rho \left(\dfrac{\partial \boldsymbol{U}}{\partial t} + \nabla \cdot (\boldsymbol{U}\boldsymbol{U}) \right) = \rho \boldsymbol{g} - \nabla p + \mu \Delta \boldsymbol{U} + \boldsymbol{F_{vol}} \\ + \, Boundary \ conditions \end{cases} \tag{1}$$

The unknowns are the velocity field $\mathbf{U} = (\mathrm{u}, \mathrm{v}, \mathrm{w})$ and the pressure p. The volumic forces are denoted $\mathbf{F_{vol}}$, $\mathbf{g}$ is the gravity acceleration, μ is dynamic viscosity coefficient and ρ is fluid density.
Although the boundary conditions are not given at this stage, we describe briefly the free surface condition because of its role in the coupling problem. The water level of the free surface (Z_s) is one of the transfered variables in the coupling algorithm.
The free surface is located at $z = Z_s(x, y, t)$ and if we consider that no particle is detached, we can write the following equations on the free surface:

$$\frac{\partial Z_s}{\partial t} - \boldsymbol{U_s} \cdot \boldsymbol{n_s} = 0 \tag{2}$$

where surface velocity field is denoted $\boldsymbol{U_s} = \mathbf{U}(\mathrm{x}, \mathrm{y}, \mathrm{Z_s}, \mathrm{t})$ and $\boldsymbol{n_s}$ is the normal defined as following:

$$\boldsymbol{n_s} = \frac{1}{\sqrt{1 + |\nabla Z_s|^2}} \left(-\frac{\partial Z_s}{\partial x}, -\frac{\partial Z_s}{\partial y}, 1\right)^T \tag{3}$$

2.1.2 Diphasic 3-D Navier-Stokes model

The volume of fluid method (V.O.F. method) was first proposed by Hirt and Nichols (Hirt et Nichols 1981) in 1981. It is an Eulerian method, which allows to follow the interface in a fixed mesh, by using a scalar indicator function, known as volume or phase fraction (α) and used to distinguish two different fluids.

In the V.O.F., the flow equations are volume averaged directly to obtain a single set of equations (eq. 2). For the phase fraction α, as the variations are:

- $\alpha = 1$: control volume is filled only with phase 1
- $\alpha = 0$: control volume is filled only with phase 2
- $0 < \alpha < 1$: interface is present

The volume fraction equation is written as:

$$\frac{\partial \alpha}{\partial t} + \nabla \cdot (\boldsymbol{U}\alpha) = 0 \tag{4}$$

Over the whole domain, it is possible to solve the Navier-Stokes equations (in both phases) by adding the surface tension and defining the quantities ρ and μ as a function of α (eq.6) .

The Navier-Stokes equations are defined in the case as follow:

$$\rho \left(\frac{\partial \boldsymbol{U}}{\partial t} + \nabla \cdot (\boldsymbol{U}\boldsymbol{U})\right) = \rho \boldsymbol{g} - \nabla p + \mu \Delta \boldsymbol{U} + \boldsymbol{F_b} \tag{5}$$

Where F_b, the surface tension force, is defined by $\boldsymbol{F}_b = \sigma \kappa \nabla \alpha$

The surface tension is denoted σ and κ is the surface curvature. ρ and μ can be then written as:

$$\begin{aligned} \rho &= \rho_1\, \alpha + \rho_2(1-\alpha) \\ \mu &= \rho_1 \alpha + \rho_2(1-\alpha) \end{aligned} \tag{6}$$

where ρ_i $(i = 1{,}2)$ and $\mu_i (i = 1{,}2)$ are respectively the density and dynamic viscosity of the i fluid.

The diphasic 3-D Navier-Stokes equations system then reads:

$$\begin{cases} \dfrac{\partial \alpha}{\partial t} + \nabla \cdot (\boldsymbol{U}\alpha) = 0 \\ \rho \left(\dfrac{\partial \boldsymbol{U}}{\partial t} + \nabla \cdot (\boldsymbol{U}\boldsymbol{U})\right) = \rho \boldsymbol{g} - \nabla p + \mu \Delta \boldsymbol{U} + \boldsymbol{F_b} \\ + \mathit{Boundary\ conditions} \end{cases} \tag{7}$$

We can report that there are many varieties of the V.O.F. method. Among the alternative method, we find the Compressive Interface Capturing Scheme for Arbitrary Meshes (CICSAM), developed by Ubbink (Ubbink 1997) and used in OpenFOAM (InterFOAM solver). The scheme allows to keep the volume fraction bounded, between 0 and 1, and get tiniest possible transition between the two fluids.

There are also VOF methods which are based on the reconstruction of the interface and its advection, for exemple the Simple Line Interface Calculation(SLIC) (Noh et Woodward 1976). It allows to have the air/water interface with precision, but these methods have a significant additional computational cost compared to the CICSAM scheme. Thereafter, we will use the VOF method implemented in InterFOAM.

2.2 Conditions at the coupling interface

A key point of the proposed method is to express the physical conditions that must be satisfied by the coupled solution at the interface between the two models. Natural conditions consist in preserving the continuity of both the velocity in water and the water level across the interface. But the diphasic model asks more information than the monophasic model can supply (air velocity for example).

That is why we introduce at the coupling interface, the restritction operator ($\mathcal{R}(.)$) and the extension operator ($\mathcal{E}(.)$), thus respectively allowing to complete coupling interface informations coming from the monophasic model (Telemac-3D) and to reduce informations coming from the diphasic model (InterFOAM). The introduction of these operators is based previous work on heterogene dimension coupling with Schwarz method, where the theoretical issues were addressed by some of the authors (Tayachi, M. 2013), (Tayachi, et al. 2014).

We can writte at the coupling interface:

$$\begin{cases} \mathbf{U_{IF}} = \mathcal{E}(\mathbf{U_{Tel}}) \\ \mathrm{Z_{sTel}} = \mathcal{R}(\mathrm{Z_{sIF}}) \end{cases} \tag{8}$$

where the diphasic model velocity and free surface are denoted respectively $\boldsymbol{U_{IF}}$, Z_{sIF} and $\boldsymbol{U_{Tel}}$ and Z_{sTel} are respectively the monophasic model velocity and free surface.

2.3 Test case description for validation

We now introduce the academic case on which the numerical experiments are performed, before going to the operational case.

2.3.1 Geometry

The computational domain is a U-shaped channel with trapezoidal section (see Figure 1). The channel is 0.79 m wide everywhere, and more than 10 m long.

The reference simulation in Figure 1 will be run in whole with the InterFOAM solver, which solves the diphasic NS3D system. The coupling model couples the monophasic NS3D for free surface flow (Telemac-3D) in the subdomain Ω_{Tel} with the diphasic NS3D equations in the subdomain Ω_{IF}. The corresponding geometry is presented in Figure 2.

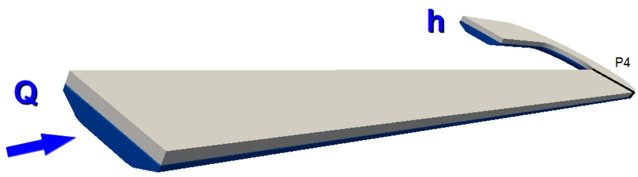

Figure 1 : Reference geometry

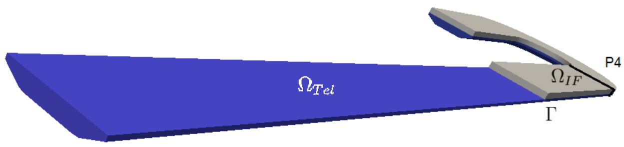

Figure 2 : Coupling geometry where Ω_{Tel} and Ω_{IF} are respectively Telemac-3D domain and InterFOAM domain, with Γ the coupling interface.

The reference simulation in Figure 1 will be run in whole with the InterFOAM solver, where the diphasic NS3D equations system is solved. The coupling model couples the monophasic NS3D for free surface flow (Telemac-3D) in the subdomain Ω_{Tel} with the diphasic NS3D equations in the subdomain Ω_{IF}. The corresponding geometry is presented in Figure 2.

2.3.2 Initial and boundary conditions

We describe here the initial and boundary conditions for the reference and coupled cases. Thereafter, we will consider only the molecular viscosity of the water $\nu = 10^{-6}$, and we will assume that we have no bottom friction.

The initial conditions for both models are $\boldsymbol{U}^0 = 0$ (fluids at rest) and constant water level $Z_s^0 = 0.114\ m$ which is imposed by a α distribution in the diphasic model.

For the reference model, the velocity is free at output, e.i., the normal derivetive of the velocity is zero. The water level is imposed $Z_{sd} = 0.114\ m$ at output . On input, we impose a constant flow rate of water ($Q_d = 0.01372\ m^3.s^{-1}$) which is distributed evenly and a null velocity in air.

Regarding the Telemac-3D/InterFOAM coupled system, we have two sub-domains Ω_{Tel} and Ω_{IF} (see Figure 2). The boundary conditions are the same than for the reference model.

2.3.3 Free surface localization uncertainty

A problem remains concerning the free surface localization. We can obtain the water level at least by two different ways. We can consider the water level as the vertical integration of α or we can consider the free surface position when the control volume contains more air phase than water phase, i.e., the free surface is when α=0.5. It could be noted that this free surface definition by integration is not valid in the case of more complex interfaces (surge, spillway, etc.).
We can see in Figure 3 that the two differents ways can give a small difference which shows well the uncertainty of the free surface localization. This uncertainty is due to the transition area of the volume fraction, which complicates the post-treatment and the coupling interface conditions.
For the post-treatement, the free surface will be calculated by vertical integration of α and the flow rate will be calculated in the following way:

$$\int_{S_{\alpha^*}} \boldsymbol{U}\, dS \tag{9}$$

where S_{α^*} is wet surface for a limit volume fraction α^* taken to 0.5.

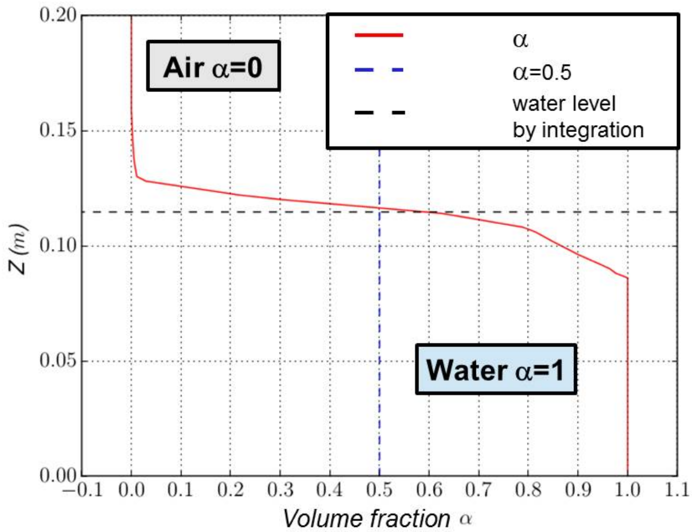

Figure 3: Vertical evolution of volume fraction (red curve) and comparison of calculation methods to find water level (blue and black dotted curves).

2.4 Coupling algorithm (Schwarz method)

The Schwarz method (Schwarz. 1869.) is a domain decomposition method. The basic principle is to reduce the resolution of a complex problem into several simpler sub-problems, thanks to an iterative algorithm. This method became popular 20 years with the development of parallel computing. The main advantages of such a coupling method is its easy implementation, and its non-intrusiveness in existing codes (only boundary conditions routines are to be changed). Its main drawback can be its computational cost, in particular when too small attention is paid to interface conditions.
The corresponding algorithm is the following:

$t = 0$, initialization step

Time loop : while $t = n\,\Delta t \leq t_{max}$

$k = 0; err = +\infty$

Schwarz loop : while err $\geq$ ε et k $\leq \mathrm{k}_{\max}$:

$$\begin{cases} \mathcal{L}_{Tel}\left(\boldsymbol{U}_{\boldsymbol{Tel}}^{n,k+1}\right) = F_{Tel}, \text{in } \Omega_{\mathrm{Tel}} \\ \boldsymbol{U}_{\boldsymbol{Tel}}{}^{n,k+1} = \boldsymbol{U}_{\boldsymbol{d}}, \text{on input} \\ \mathrm{Z}_{\mathrm{sTel}}^{\mathrm{n,k+1}}(\Gamma) = \mathcal{R}\left(\alpha_{\mathrm{IF}}^{\mathrm{n,k}}\right), \text{on interface} \end{cases}$$

Time sub – loop : at t to $(\mathrm{n}+1)\Delta\mathrm{t}$ where $t' = \tilde{n}\,\Delta t' + t$:

$$\begin{cases} \mathcal{L}_{IF}\left(\boldsymbol{U}_{\boldsymbol{IF}}^{\tilde{n},k+1}\right) = F_{IF}, \text{in } \Omega_{\mathrm{IF}} \\ Z_{sIF}^{\tilde{n},k+1} = Z_{sd}\ , \text{on outup} \\ \boldsymbol{U}_{\boldsymbol{IF}}\,(X_o)^{\tilde{n},k+1} = \mathcal{E}(\boldsymbol{U}_{\boldsymbol{Tel}}), \text{on interface} \end{cases}$$

$\tilde{n} \leftarrow \tilde{n} + 1$

Sub – loop end

Calculate convergence criterion err

$k \leftarrow k + 1$

Shwarz loop end

$n \leftarrow n + 1$

Time loop end

Algorithm 1: Schwarz based coupling algorithm for coupled Telemac-3D and InterFOAM

At convergence, we should have:

$$\begin{cases} \boldsymbol{U}_{\boldsymbol{IF}} = \begin{cases} \boldsymbol{U}_{\boldsymbol{Tel}}\,, & z \leq Z_{sTel} \\ \overline{\boldsymbol{U}_{\boldsymbol{air}}}\boldsymbol{n}(1-\alpha) + \alpha\,\boldsymbol{U}_{\boldsymbol{Tel}}\,, & z > Z_{sTel}\ 0 \leq \alpha < 1 \end{cases} \\ Z_{sTel} = Z_{sIF} = \displaystyle\int_{Z_{min}}^{Z_{max}} \alpha\, dz \end{cases} \tag{10}$$

where Z_{sIF}, Z_{sTel} and $\boldsymbol{U}_{\boldsymbol{IF}}, \boldsymbol{U}_{\boldsymbol{Tel}}$ are respectively the water level and the velocity for InterFOAM and Telemac-3D, along the coupling interface.

For air phase, in the diphasic model, a mean velocity is imposed $\overline{\boldsymbol{U}_{\boldsymbol{air}}}\boldsymbol{n}$ which is distributed evenly along the coupling interface. Furthermore, the vertical integration of α is our restriction operator ($\mathcal{R}$) and the selected distribution of velocity on coupling interface is our extension operator ($\mathcal{E}$).

It should be noted that the numerical diffusion of the free surface position implies an uncertainty in imposing the velocity in Ω_{IF} coming from Ω_{Tel}. In fact, part of the velocity is injected in the transition area of the volume fraction, involving also an uncertainty in the flow rate which is imposed at the coupling interface, since the velocities are actually a mixture of air and water velocities. Furthermore, the imposed velocity in the transition area of volume fraction plays also a role on friction between air and water at the fluids interface. However, in view of the boundary conditions which can be used in InterFOAM, we don't yet have a better solution to limit the flow rate loss and to keep a good convergence.

Finnally, the convergence criterion (err) is chosen as follows:

$$err = \max\left[\left|Z_{sTel}^{k} - Z_{sTel}^{k+1}\right|\right] + \max\left[\left|\boldsymbol{U}_{\boldsymbol{IF}}^{\boldsymbol{k}} - \boldsymbol{U}_{\boldsymbol{IF}}^{\boldsymbol{k+1}}\right|\right] \tag{11}$$

where the water level and the velocity are taken at the coupling interface. This criterion imposes a simultaneous convergence between the water levels and the velocities and has been chosen equal to 10^{-4}, which is here an acceptable precision.

2.5 Numerical results

In this section, we will compare the reference model with the coupled model, allowing to see the coupling limits and the treatment of volume fraction transition area.
In view that we aren't taking into account the air friction on the free surface in Ω_{Tel} and that we inject the velocity coming from Ω_{Tel} in the transition area of the volume fraction, we are therefore waiting for a flow rate loss in Ω_{IF} and an adjusting flow area after the coupling interface. This adjusting flow area can be seen in Figure 4.

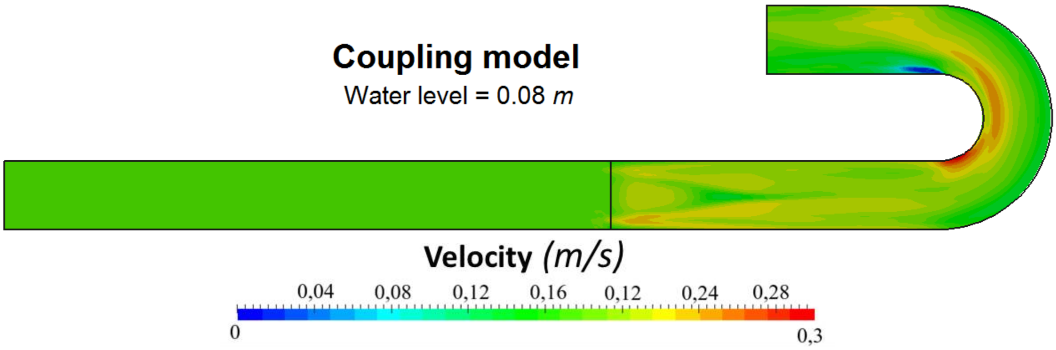

Figure 4 : Horizontal section of the velocity at the water level 0.08 m, when the flow is established

Furthermore, we plot the relative error with the reference for the free surface and the flow rate. The flow rate after coupling interface has a 4% loss in Ω_{IF} at P4 position (Figure 5), when the flow is established, i.e., after 400s. The differences on the free surfaces in the two sub-domains and on the flow rate in Ω_{Tel}, are lower when the flow is established (less than 1%), such as we can see Figure 6 for free surface in Ω_{IF} at P4 position.
These observed differences are due to our choice for the treatment of volume fraction transition area, seen as earlier(§2.4).
We observe also a difference on the velocity profil (Figure 7), alos coming from the treatment of the volume fraction transition area.

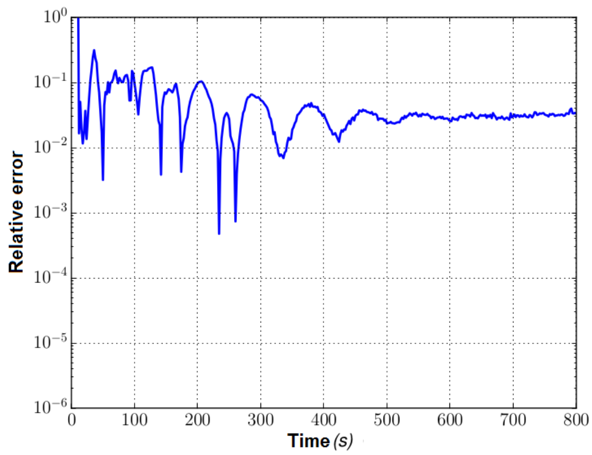

Figure 5 : Flow rate relative error between reference and couplied model at position P4

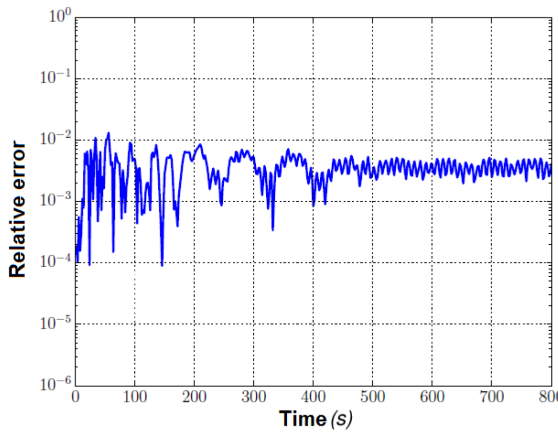

Figure 6 : free surface relative error between reference and coupled model at channel center in P4

To reduce the injected Ω_{Tel} velocity impact in the volume fraction transition area in Ω_{IF}, we modify the interface conditions to force the reduction of the transition area of volume fraction at coupling interface position, i.e., we impose $\alpha = 1$ if $\alpha > 0.55$ and we impose $\alpha = 0$ if $\alpha < 0.45$ (this deliberately drastic reduction allowing thus to reduce the variability area of α).
By reducing the interval $\alpha(Z_{sTel}) < \alpha(z) < 1$, we should have a better transmission of quantities between both sub-domains in the water phase. This should reduce flow rate differences and velocity profile differences.
It should be noted that this approximation actually results in minor changes of water level at the coupling interface. Moreover, it inuces stability issues, requiring a smaller time step to converge.

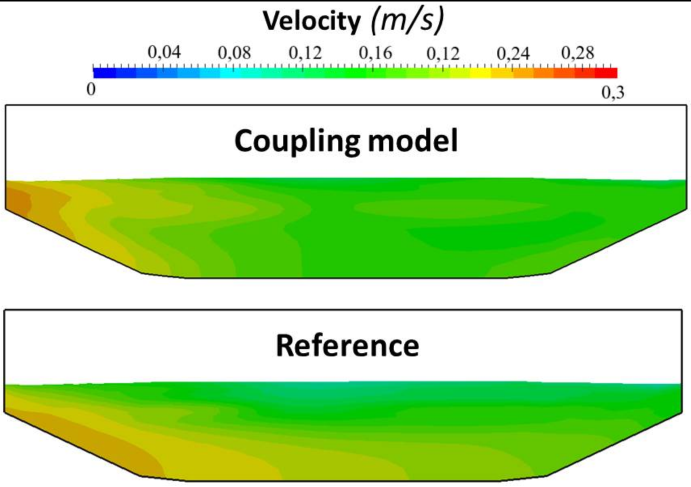

Figure 7 : Velocity profile comparison between reference model and coupled model at the P4 position

As can be seen in Figure 8 and Figure 9, the velocity profile and flow rate are more similar with the reference model than with the previous coupling. This difference illustrates the impact of the interface condition. However for the operational case, we will use the interface condition presented in the section 2.4 since this second interface condition doesn't allow the simulation convergence.

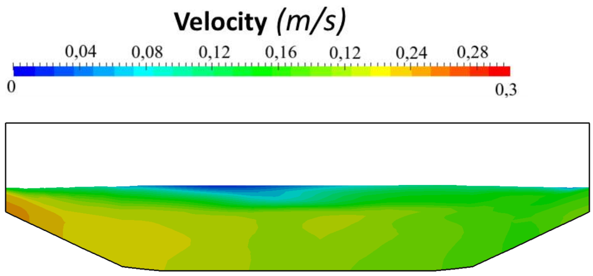

Figure 8 : Velocity profiles of coupled model with the reduced volume fraction area at P4 position.

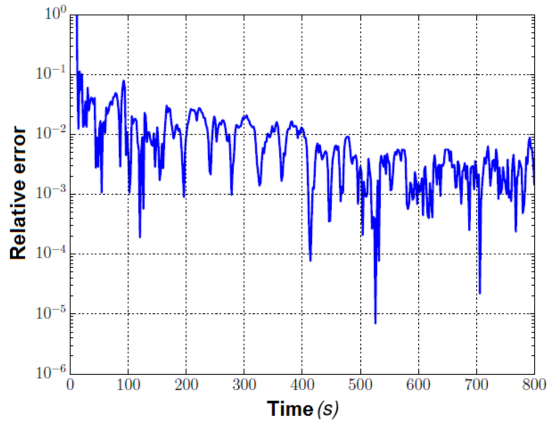

Figure 9 : Flow rate relative error between reference and coupled model with the reduced volume fraction area at position P4

3 OPERATIONAL APPLICATION

3.1 The Rusumo project

This case study corresponds to a future hydroelectric plant located at the Rusumo Falls, at the border between Tanzania and Rwanda on the Kagera River.

The application objective is to allowing to study the whole domain (Figure 10), from the water intake dam to the hydroelectric plant where the pipes are.

We model indeed the hydroelectric plant with InterFOAM since Telemac-3D can't resolve in charge flow. Moreover, the remaining domain is modeled with Telemac-3D, allowing to reduce the computation cost which would be much too large if resolved with InterFOAM. The whole domain resolution is also only possible with couplied models.

3.2 Geometry and mesh

In Figure 11 , we can see the geometry . The position of the dam, the channel, the water intake and the hydroelectric plant are indicated on Figure 10.

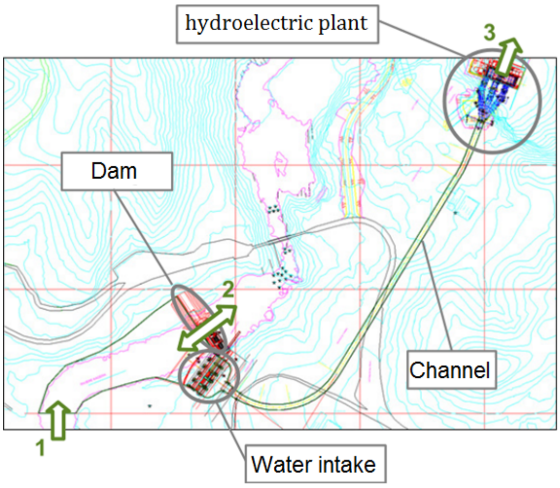

Figure 10 : Map from water intake dam to hydroelectric plant

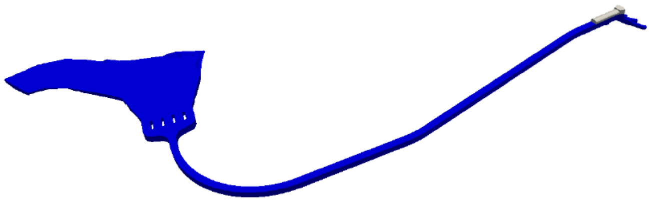

Figure 11 : Coupled models geometry

The mesh in sub-domain Telemac-3D (Ω_{Tel}, composed of river, water intake and channel) has approximately 63 200 triangle elements (Figure 12) duplicated on 4 plans in the vertical (resulting in more than 132 500 nodes). In sub-domain InterFOAM (Ω_{IF}, composed of hydroelectric plant, Figure 13) we have more than 233 000 elements which are mainly hexahedrals and polyhedrals (approximately 600 400 nodes).

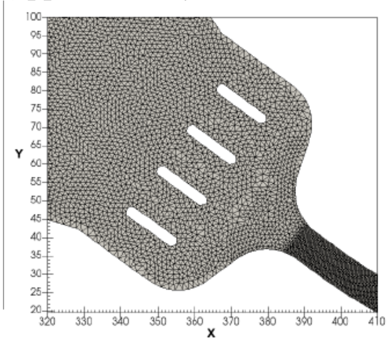

Figure 12 : Example of Telemac-3D mesh at the water intake

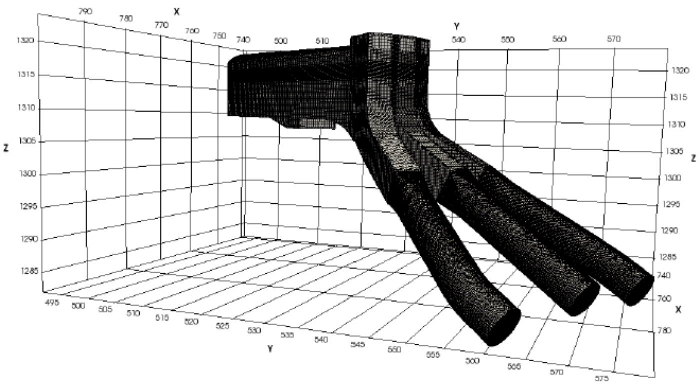

Figure 13 : Sub-domain InterFOAM mesh.

3.3 Initial and boundary conditions

We describe here the initial and boundary conditions for this operational case. The initial conditions can be seen one Figure 14. We impose a water level and velocity in the both sub-domains.
In Table 1, we can see the upstream and downstream conditions. Furthermore, we take into account the turbulence of the flow and the friction for the wall.

	Localisation of Figure 10	Boundary types	Values
Rivers	1	Flow rate	$400\ m^3.s^{-1}$
Spillway (dam)	2	Water level	$1320\ m$
Pipes output	3	Flow rate	$120\ m^3.s^{-1}$ for one pipe

Table 1 : Boundary conditions of the operational case

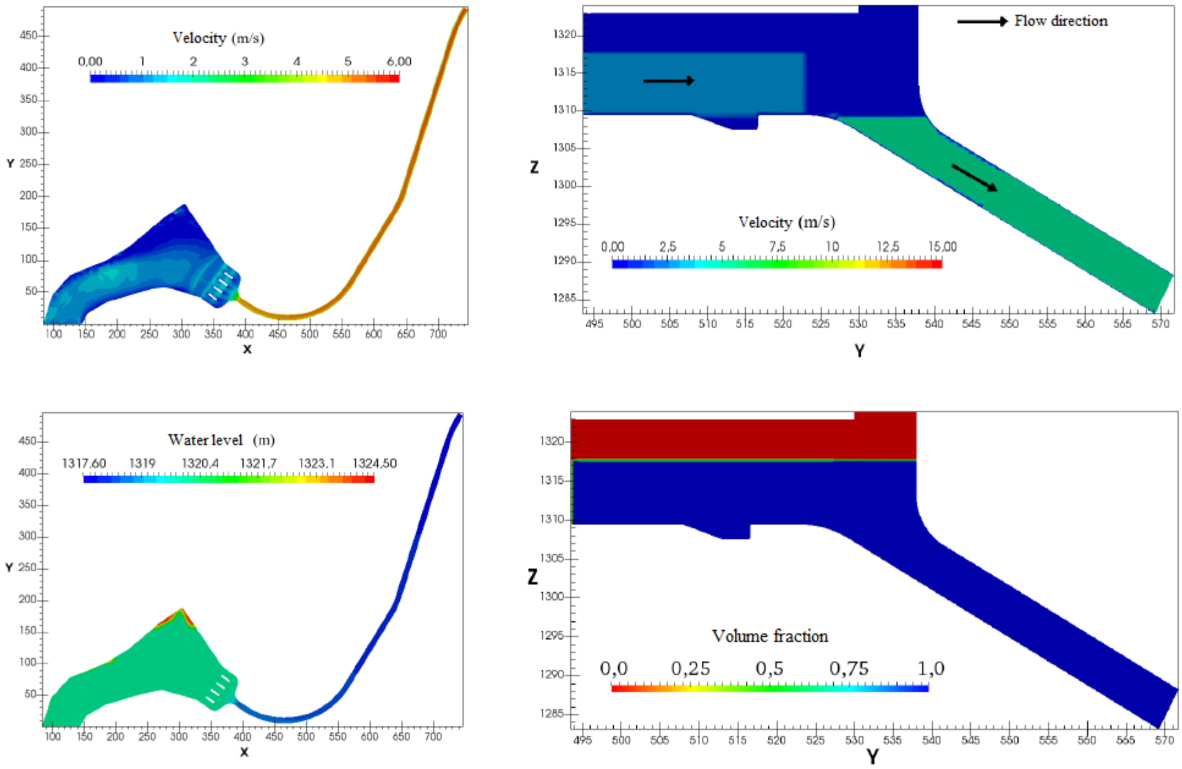

Figure 14: Initial conditions of velocity and water level for Telemac-3D sub-model (left) and for InterFOAM sub-model (right). The water level in sub-model InterFOAM is of 1317.75 m.

3.4 Interface conditions

The interface conditions are the same that in equation (eq. 10). But this time, we imposed $\overline{\boldsymbol{U_{air}}} = 2\ m.s^{-1}$ to avoid an air recirculation in the channel.

Furthermore, an instability can occur at the coupling interface which could be a problem for the convergence. That's why we imposed an initial velocity of the water phase in InterFOAM sub-domain to reduce the transient state which results to the establishment of the flow.

3.5 Analysis of the results

A physical model of the hydroelectric plant area was built, where we observed a lot of turbulence, vortex, and air aspirations near and in the pipes. During the comparison (Figures 15 and 16) between numerical model and physical model, we find well similar behaviours when the flow is established.

We can also see a water level difference (Figure 16) between numerical model and physical model due to the draining effect, at the beginning of the simulation. The draining effect in Ω_{IF}, during the transitory state is due to two reason. The first is water level increase and velocity decrease near coupling interface in Ω_{Tel} spreading to river, causing a flowrate decrease at the coupling interface. The second is a flowrate loss at coupling interface like we have seen in §2.4 and §2.5. However, this draining effect in Ω_{IF} is compensated, during the rest of the simulation with the decrease water level and the velocity increase (the hydroelectric plant remains under the river level), allowing to have a flowrate balanced (input/output) in sub-domain Ω_{IF}, when the flow is established.

Figure 15 : Comparison between the numerical model (left picture) and the physical model (right picture). The drawn free surface is for $\alpha = 0.5$.

With Schwarz algorithm, the iteration number remains between 4 and 5 iterations. Overall, the results are satisfactory (in particular the swirl pattern and air aspirations in the pipes). Although the computational costs are significant, we are able to solve the whole domain (from the river to the plant) in 3-D thanks to this Telemac-3D / InterFOAM coupling. Before these developments, the whole domain could only be modeled with a physical model, at least in Artelia.

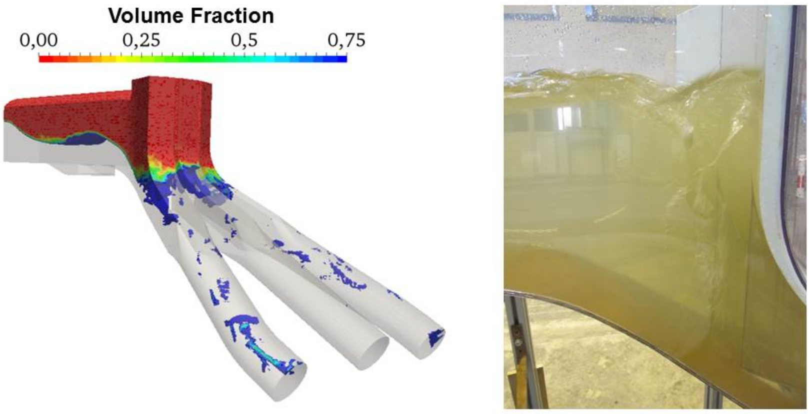

Figure 16 : Comparison between the numerical model (left picture) and the physical model (right picture) where we can see air aspiration.

4 CONCLUSION

The main objective of this paper was to propose algorithms to couple a monophasic model and a diphasic model in the context of river modeling with the Telemac-3D and InterFOAM solvers.
This coupling allows to perform a first study of an operational case, such as a water intake dam and a hydroelectric plant, where it's not possible resolve with only Telemac-3D (which doesn't model pipes) or InterFOAM (too expensive cost). For this, we developped a Schwarz method which was assessed in an first test case, namely a U-shape channel. In this simple test case, we showed the difficulties due to the uncertainty on the exact position of the free surface, even if performances are quite satisfactory. More specifically, we saw, in this test-case, the significance of the treatment of α transition area at coupling interface, which can add a velocity diffusion in α transition area (flowrate loss) and can change air/water frictions. But thanks to the developped coupling method, it is possible the whole domain of Rusumo project, where our numerical results exhibit the same behaviour than the physical model, when the flow is established.
The next step for this work will be to reduce computation costs which remain significant. It would also be interesting to study new coupling interface conditions. For this, a study of "absorbing boundary conditions" could be interesting. Last, different VOF methods could be studied like VOF-SLIC method where the uncertainty of the free surface is pulled out, but the computation cost should be more important.

BIBLIOGRAPHIE

Hervouet, J-M. «Hydrodynamics of free surface flows, modelling with the finite element method.» *John Wiley & Sons.*, 2007.

Hirt, C., et B.D. Nichols. «Volume of Fluid (VOF) method for the dynamics of free boundaries.» *Journal of Computational Physics* 39(1) (1981): 201–225.

Noh, W. F., et P. Woodward. «SLIC (Simple Line Interface Calculation).» *In Proceedings of the Fifth International Conference on Numerical* (In S. B. Heidelberg., editor), June 28–July 2 1976: 330–340.

Schwarz., H.A. «Über einige abbildungsaufgaben.» *Journal für die reine und angewandte Mathematik.* 11 (1869.): 65–83.

Tayachi, M. «Couplage de modèles de dimensions hétérogènes et application en hydrodynamique.» *Ph.D thesis, Université de Grenoble*, 2013.

Tayachi, M., A. Rousseau, E. Blayo, N. Goutal, et V. Martin. «Design and analysis of a Schwarz coupling method for a dimensionally heterogeneous problem.» *International Journal for Numerical Methods in Fluids*, 2014: 75(6), 446-465.

Ubbink, O. *Numerical prediction of two fluid systems with sharp interfaces.* PhD thesis, Imperial College of Science, 1997.